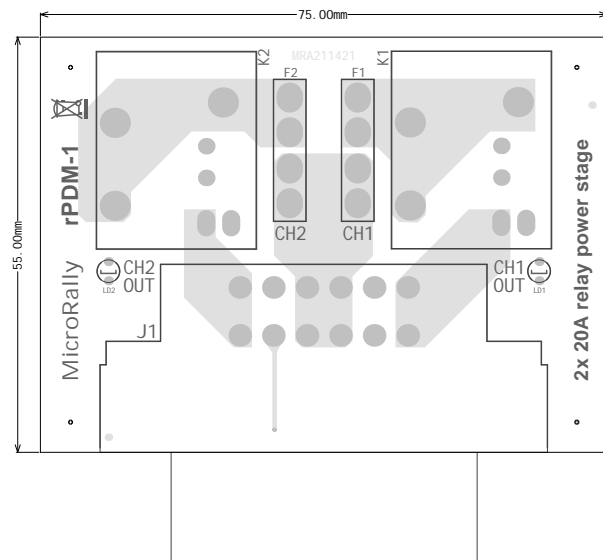


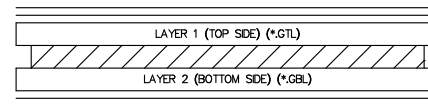
GERBER NOTES



Symbol	Count	Hole Size	Plated	Hole Type	Via/Pad
◇	1	0.300mm	PTH	Round	Via
⊠	4	0.750mm	PTH	Round	Pad
▽	4	1.300mm	PTH	Round	Pad
○	4	1.300mm	PTH	Slot	Pad
⊗	12	1.700mm	PTH	Round	Pad
□	6	2.400mm	PTH	Round	Pad
⊛	8	2.500mm	PTH	Round	Pad
⊕	4	3.200mm	NPTH	Round	Pad
	43 Total				

Slot definitions : Routed Path Length = Calculated from tool start centre position to tool end centre position.
Hole Length = Routed Path Length + Tool Size = Slot length as defined in the PCB layout

1.0 OZ L1
1.50 mm Core - FR4
1.0 OZ L2



SILKSCREEN (TOP SIDE) (*GTO)
SOLDERMASK (TOP SIDE) (*GTS)
1.6 mm +/- 10% LAYER DETAIL
2 LAYERS
SOLDERMASK (BOTTOM SIDE) (*GBS)
SILKSCREEN (BOTTOM SIDE) (*GB0)
PASTE (BOTTOM SIDE) (*GBP)

NOTES: (UNLESS OTHERWISE SPECIFIED)

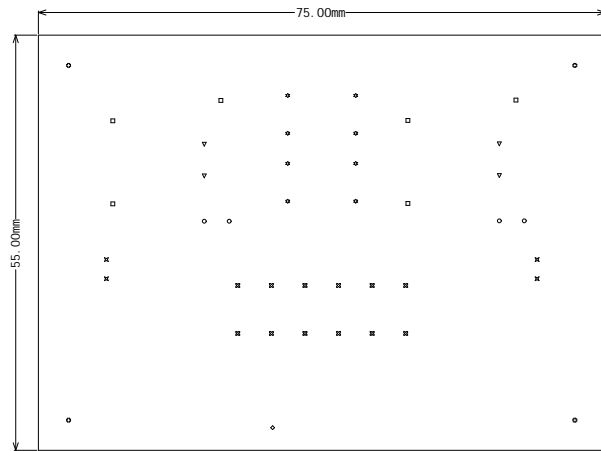
- THIS IS 2 LAYER BOARD
- MATERIAL: FR4, TG 150 DEGREE C MIN
- FR4 DIELECTRIC CONSTANT NOT SPECIFIED
- FINISHED BOARD THICKNESS TO BE 1.60MM +/- 10%
- TRACE WIDTHS IN ARTWORK ARE FINISHED SIZES
- SEE FILM FOR LAYER SEQUENCE AND COPPER THICKNESSES (SHOWN BEFORE PLATING)
- MIN TRACE/SPACE 0.60/0.25MM
- SEE DRILL CHART FOR FINISHED HOLE SIZES
- MIN DRILL 0.75MM
- HOLE TOLERANCE IS +/-3MIL UNLESS OTHERWISE SPECIFIED
HOLE COPPER THICKNESS MIN 0.7MIL
SLOT TOLERANCE +/-0.1MM
BORDER OUTLINE TOLERANCE +/-0.15MM
- SURFACE PLATING: HASL, Pb FREE
- SOLDERMASK: LPI, BOTH SIDES. COLOR GREEN
- SILKSCREEN: TOP AND BOTTOM SIDE. COLOR WHITE
- ALL BOARDS MUST BE ELECTRICALLY TESTED FOR ISOLATION (SHORTS) AND CONTINUITY (OPENS)

Project: rPDM-1	
Author: Andis Jargans	Revision: 1
Date: 11.04.2021	Size: A4
File: rPDM-1_r1.PcbDoc	MicroRally

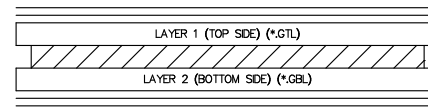
GERBER NOTES

Symbol	Count	Hole Size	Plated	Hole Type	Via/Pad
◇	1	0.300mm	PTH	Round	Via
✕	4	0.750mm	PTH	Round	Pad
▽	4	1.300mm	PTH	Round	Pad
○	4	1.300mm	PTH	Slot	Pad
⊗	12	1.700mm	PTH	Round	Pad
□	6	2.400mm	PTH	Round	Pad
⊛	8	2.500mm	PTH	Round	Pad
⊕	4	3.200mm	NPTH	Round	Pad
	43 Total				

Slot definitions : Routed Path Length = Calculated from tool start centre position to tool end centre position.
 Hole Length = Routed Path Length + Tool Size = Slot length as defined in the PCB layout



1.0 OZ L1
 1.50 mm Core - FR4
 1.0 OZ L2



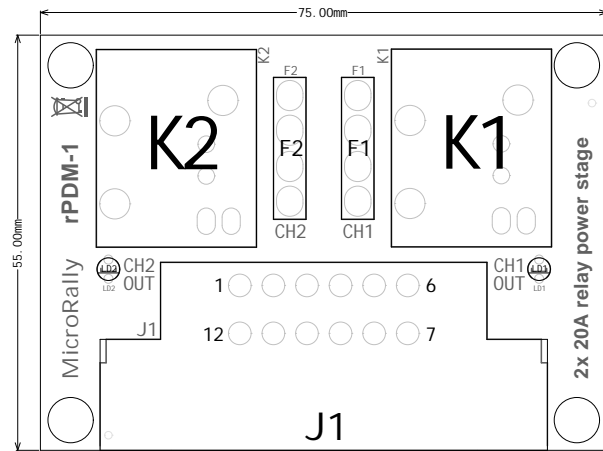
SILKSCREEN (TOP SIDE) (*GTO)
 SOLDERMASK (TOP SIDE) (*GTS)
 1.6 mm +/- 10% LAYER DETAIL
 2 LAYERS
 SOLDERMASK (BOTTOM SIDE) (*GBS)
 SILKSCREEN (BOTTOM SIDE) (*GB0)
 PASTE (BOTTOM SIDE) (*GBP)

NOTES: (UNLESS OTHERWISE SPECIFIED)

- THIS IS 2 LAYER BOARD
- MATERIAL: FR4, TG 150 DEGREE C MIN
- FR4 DIELECTRIC CONSTANT NOT SPECIFIED
- FINISHED BOARD THICKNESS TO BE 1.60MM +/- 10%
- TRACE WIDTHS IN ARTWORK ARE FINISHED SIZES
- SEE FILM FOR LAYER SEQUENCE AND COPPER THICKNESSES (SHOWN BEFORE PLATING)
- MIN TRACE/SPACE 0.60/0.25MM
- SEE DRILL CHART FOR FINISHED HOLE SIZES
- MIN DRILL 0.75MM
- HOLE TOLERANCE IS +/-0.3MIL UNLESS OTHERWISE SPECIFIED
 HOLE COPPER THICKNESS MIN 0.7MIL
 SLOT TOLERANCE +/-0.1MM
 BORDER OUTLINE TOLERANCE +/-0.15MM
- SURFACE PLATING: HASL, Pb FREE
- SOLDERMASK: LPI, BOTH SIDES. COLOR GREEN
- SILKSCREEN: TOP AND BOTTOM SIDE. COLOR WHITE
- ALL BOARDS MUST BE ELECTRICALLY TESTED FOR ISOLATION (SHORTS) AND CONTINUITY (OPENS)

Project: rPDM-1	
Author: Andis Jargans	Revision: 1
Date: 11.04.2021	Size: A4
File: rPDM-1_r1.PcbDoc	MicroRally

ASSEMBLY NOTES



OUT2	IN2	VBAT	VBAT	IN1	OUT1
OUT2	IGN	VBAT	VBAT	GND	OUT1
J1 PINOUT					

Project: rPDM-1	
Author: Andi s Jargans	Revi si on: 1
Date: 11.04.2021	Si ze: A4
Fi le: rPDM-1_r1.PcbDoc	Mi croRal ly

MicroRally

rPDM-1



CH2
OUT
LD2

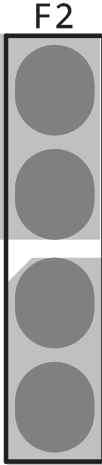
J1

CH1
OUT
LD1

K2

MRA211421

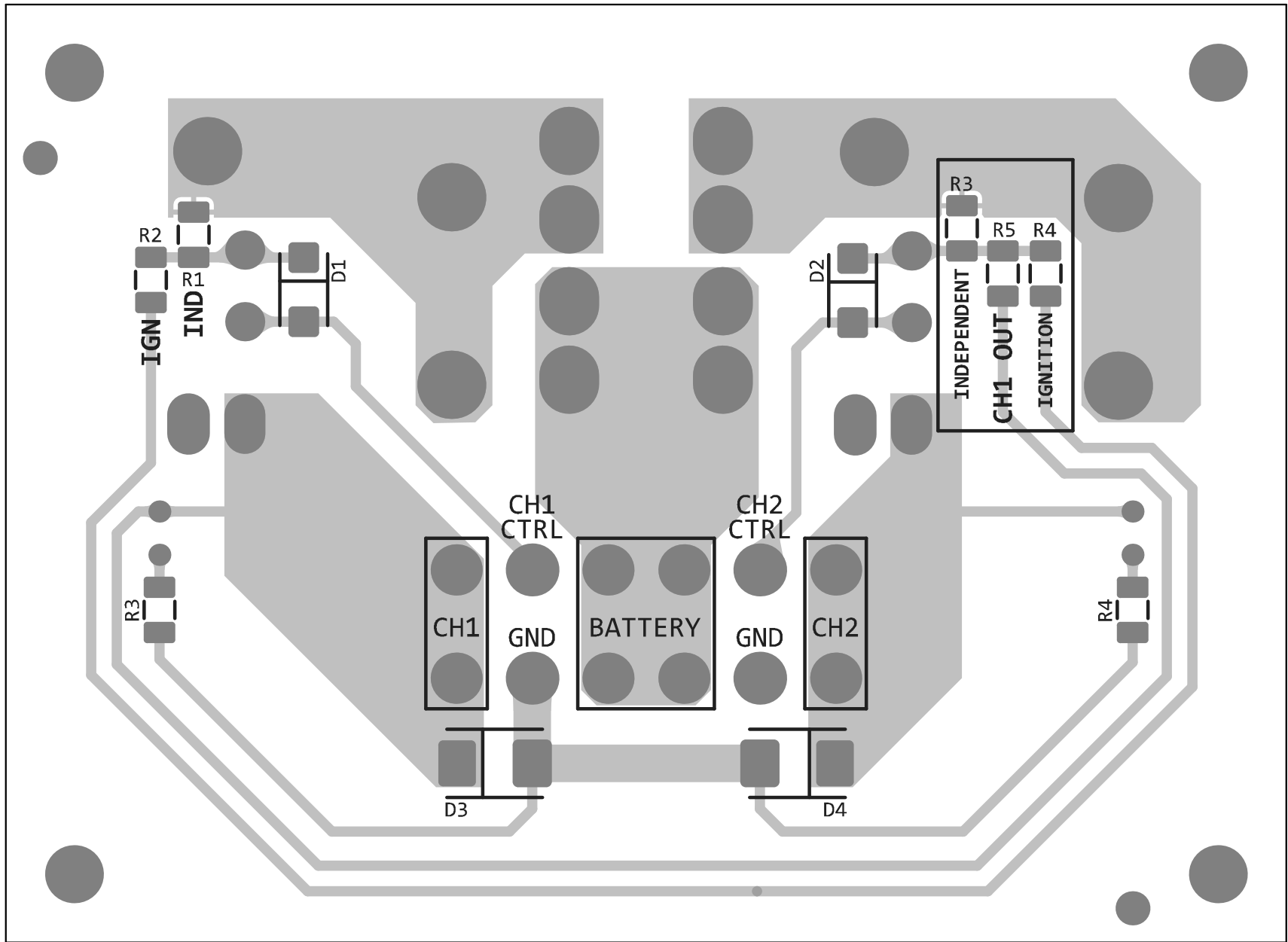
K1

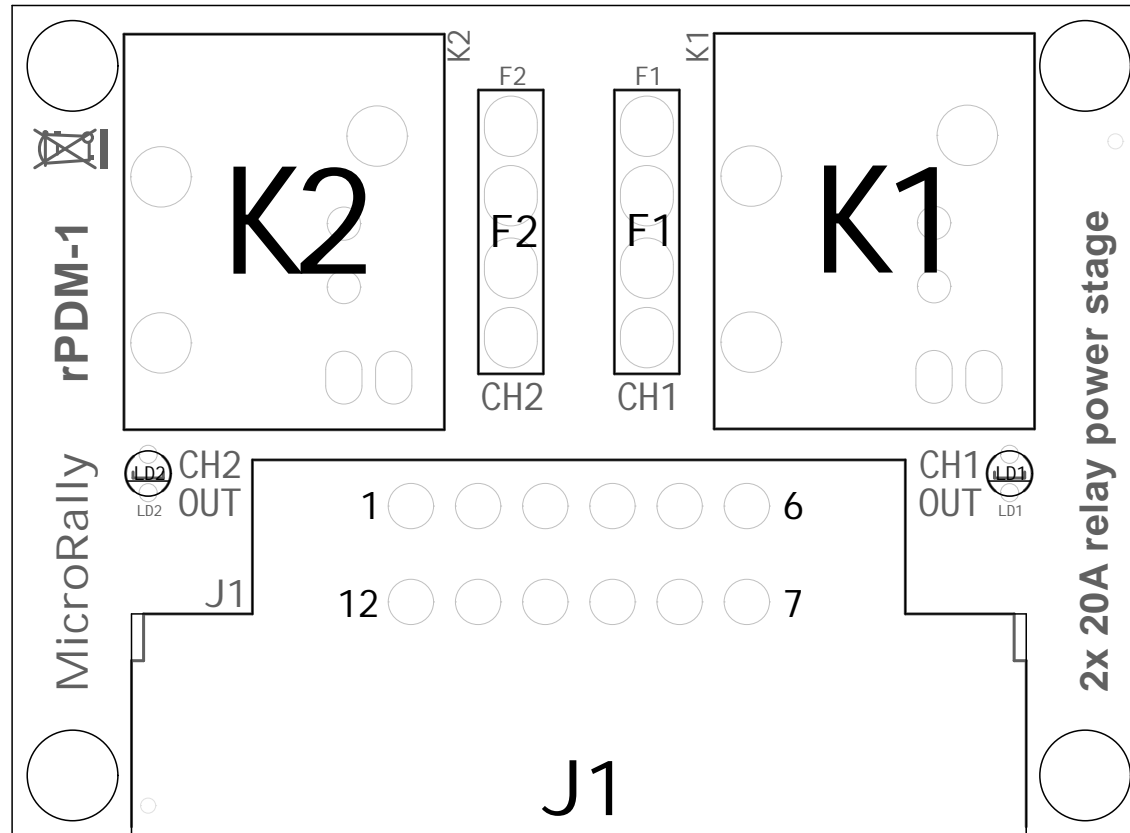


CH2

CH1

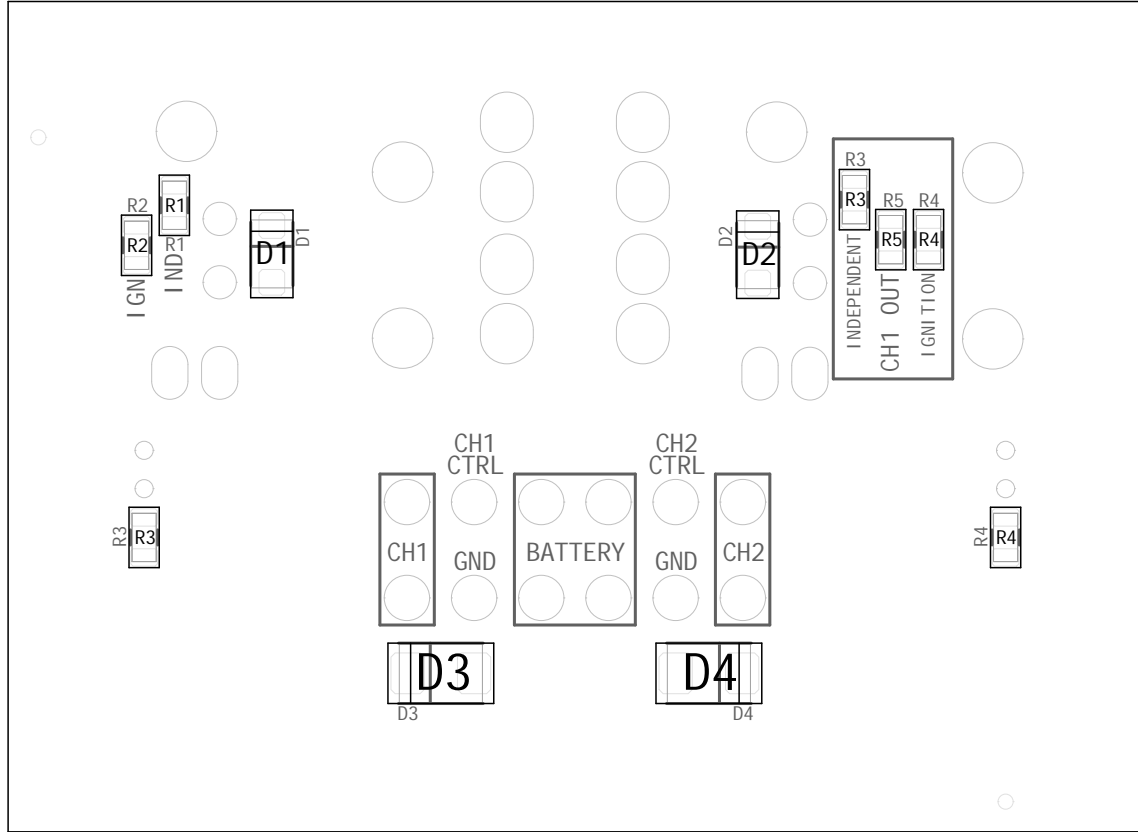
2x 20A relay power stage



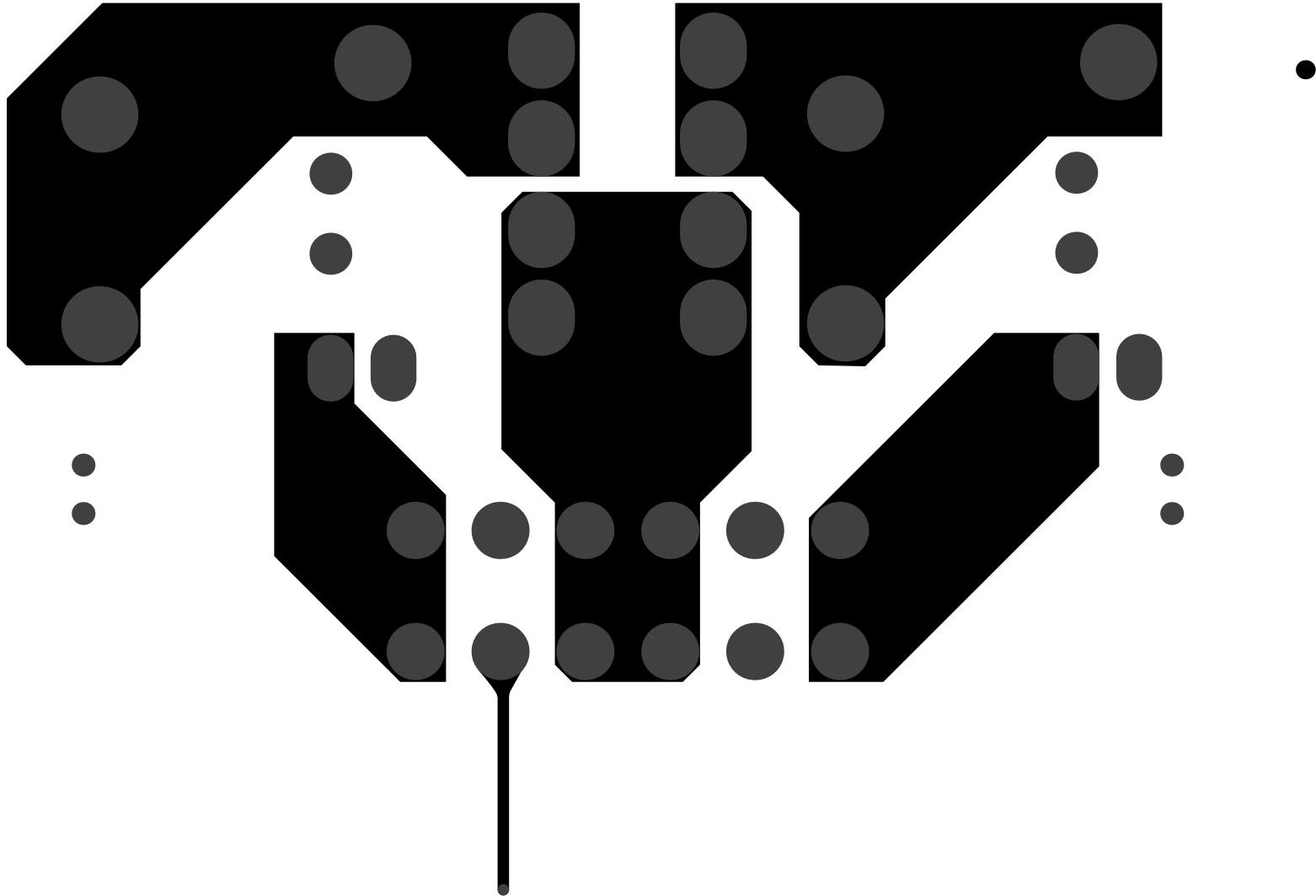


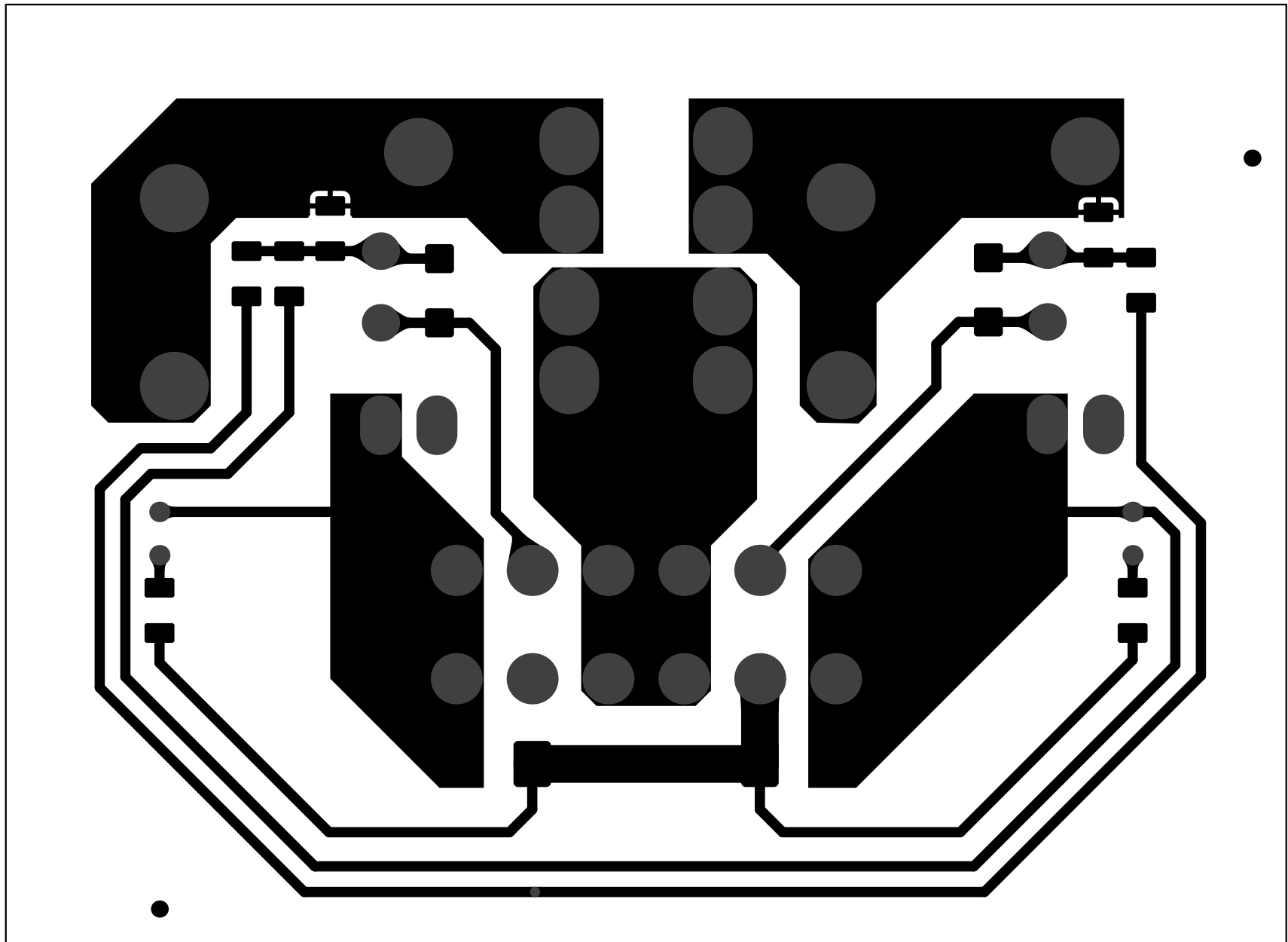
OUT2	IN2	VBAT	VBAT	IN1	OUT1
OUT2	IGN	VBAT	VBAT	GND	OUT1

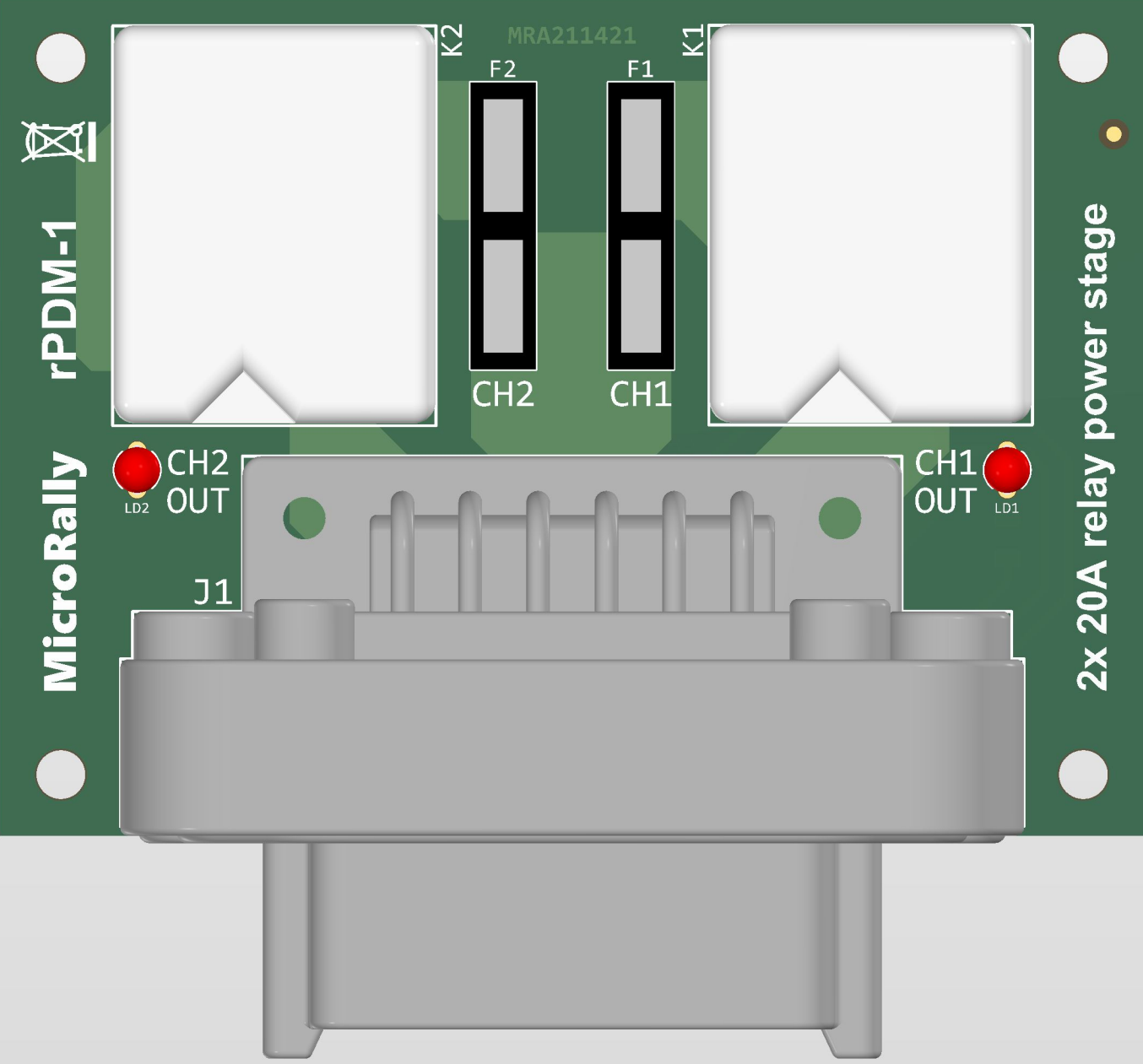
J1 PINOUT



MRA211421







MicroRally

rPDM-1



CH2
OUT

J1

K2

F2

CH2

MRA211421

F1

CH1

K1

CH1
OUT



2x 20A relay power stage

